Magnetic Bearing Lab # 2: Magnetic Bearing System Identification*

Nancy Morse, Roy Smith and Brad Paden
e-mail: nancy@me-server.ucsb.edu

May 29, 1996

Experimental determination of a system model is an important part of the modeling process. This phase of controller design is referred to as *system identification*. In the lab that follows, we will use a Hewlett Packard 3562A Dynamic Signal Analyzer to collect experimental data from the MBC 500 Magnetic Bearing system. Specifically, we will use the signal analyzer's swept sine function to experimentally determine the transfer function of a single input/ single output (SISO) path through the magnetic bearing system. In the following discussion we will describe the dynamic signal analyzer, its swept sine function, and the collection of frequency response data from the magnetic bearing system. After transferring the data from the signal anayzer to MATLAB, we will use some MATLAB routines to find a suitable system model.

2.1 The HP 3562A Dynamic Signal Analyzer

The dynamic signal analyzer is a device which aids in the experimental identification of linear systems by simultaneously injecting a signal into the system and measuring the system response. The signal analyzer can then process the recorded data to generate both visual and mathematical representations. A general description of the analyzer front panel and the basic functions of the keys and input/output connections we will be using follows. Refer to the HP 3562A Operator's Manual for detailed descriptions of the signal analyzer functions.

---

*The following is a list of the hardware and software required to perform this lab. Hardware: MBC 500 Magnetic Bearing, Hewlett Packard 3562A Dynamic Signal Analyzer, Personal Computer, IEEE 488 Port Card, Parallel cable, Various coaxial cables and adaptors. Software: MATLAB, μ-TOOLS, HP Standard Data Format Utilities and IEEE 488 Port Card Driver.

MEASUREMENT The measurement keys are used to select the general function, such as swept sine, which the analyzer will be performing. In addition, these keys are used to choose the "Source" signal amplitude and frequency range.

DISPLAY These keys are used to select the information to be plotted on the display window. The window can be broken up into two parts (parts A and B) if multiple plots are desired. For example, one can display the magnitude Bode plot of the data on the upper part and the phase Bode plot on the lower part.

CONTROL The control keys allow the user to start a measurement. They also allow the user to pause and continue a given measurement.

HP-IB These keys allow the user to save the data to a hard drive for transfer to other software environments or for later processing on the signal analyzer.

MARKERS The marker keys allow the user to obtain detailed information about specific points on the displayed plots. The circular knob can be turned to move the marker from one frequency to the next on the plot.

SOURCE This output port is used to drive the system under test with a signal generated by the signal analyzer.

CHANNELS 1 & 2 These are two input ports available on the signal analyzer for recording signal information from the system under test. For determining a system transfer function, "Channel 1" is considered the system input signal and "Channel 2" is treated as the system output signal.

For this lab, we will be using primarily the swept sine function on the signal analyzer. The swept sine function drives the output port labled "Source" on the signal analyzer with a sequence of sinusoids each with a different frequency. The input ports, "Channels 1 and 2", are then used to record the system response and determine the transfer function of the system under test. During a swept sine test, "Channel 1" is treated as the system input and "Channel 2" as the system output. The user may select the sinusoidal frequency range and sweep direction, i.e. ascending or descending. The amplitude of the sinusoids may also be

chosen. This amplitude can be changed during a test by pausing the sweep, adjusting the amplitude, and continuing the sweep again. This process may be necessary to avoid exciting certain resonances within the system.

The level of the "Source" signal amplitude will not affect the transfer function calculated by the signal analyzer for systems which are linear and are unaffected by noise. However, most physical systems are non-linear and noisy. For such systems, a low signal amplitude will cause a low signal-to-noise ratio in the system response leading to transfer function estimates with a lot of variance (i.e. The estimates are noisy.). High signal amplitudes can saturate the system under test yielding corrupted data and can also increase the significance of the non-linearities in the system. Thus, there is a trade-off in the selection of sinusoidal amplitude.

2.2   Setting Up the Magnetic Bearing for Testing

The magnetic bearing system configuration is as shown on the magnetic bearing front panel block diagram and in Figure 1. The bearing system contains four on-board controllers which levitate the bearing.
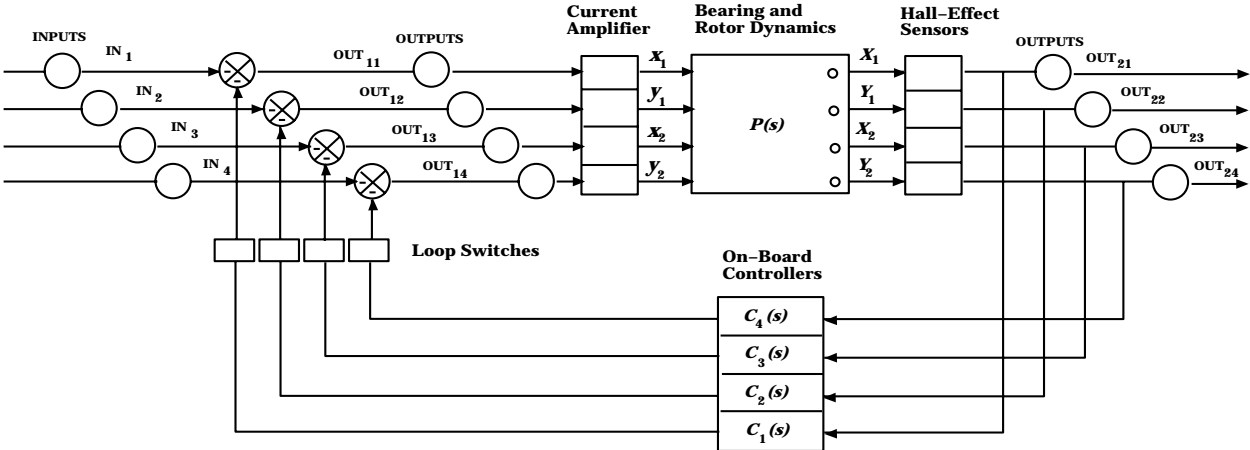

Figure 1: Magnetic Bearing Block Diagram

As we saw in *Magnetic Bearing Lab # 1*, the magnetic bearing is an unstable system. If any of the on-board controllers are disconnected, the rotor will hit the touch-down bushings in the bearing housings which form a physical limit for the rotor. This can be observed by disconnecting one controller at a time, while the unit is on, and noticing the effect. Because

3

of this instability, any data obtained from the bearing for the purpose of determining its transfer function must be taken while the bearing is controlled. In other words, we must perform closed-loop system identification instead of more direct open-loop system identification techniques.
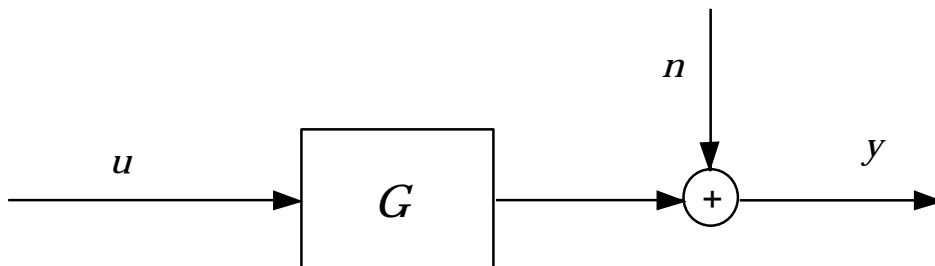

Figure 2: Magnetic Bearing Block Diagram

One of the primary challenges of system identification is to estimate the system transfer function in the presence of noise. Suppose we wish to identify the system G shown in Figure 2. If G is stable, we may perform open-loop system identification. If we apply an input u and record the response y, we have the following relationship:

$$y = Gu + n.$$

Dividing y by u gives

$$\frac{y}{u} = G + \frac{n}{u}$$

which can be used to estimate G. Typically, the output data y is collected over an extended period of time and the response y/u is averaged to determine an approximate transfer function for the system. This process is valid in the open-loop case where u is uncorrelated with n. Since for most systems n has zero mean, the term n/u will average to zero, and its effect on the average y/u will be minimal leading to the desired transfer function approximation

$$G \cong \frac{y}{u}.$$

If, however, u were correlated with n, the term n/u would not average to zero and would add to and corrupt the estimate of G.

For the closed-loop system configuration shown in Figure 3, the relationship $y/u = G + n/u$ again holds; however, the input u can be expressed as follows:

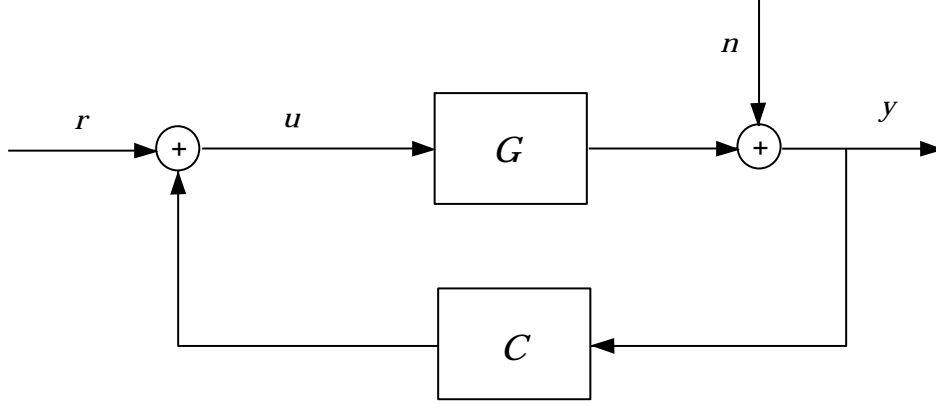$$u = \frac{1}{1 + GC}r - \frac{C}{1 + GC}n.$$

4

Figure 3: Magnetic Bearing Block Diagram

From this equation, it is clear that u and n are correlated, and therefore the term n/u will not average to zero as y/u is averaged. Since the effect of the noise on the system response cannot be neglected, we would like to see what effect the noise will have on our transfer function estimate. We can express y/u in the following way:

$$\frac{y}{u} = \frac{n + Gr}{r - Cn}.$$

Notice that as n becomes small compared to r, we have

$$\frac{y}{u} \cong G.$$

However, if n is large compared to r, the following holds:

$$\frac{y}{u} \cong -\frac{1}{C}.$$

The estimate we obtain from y/u will be somewhere in between G and $-1/C$; however, without exact knowledge of n, we cannot solve for G. Therefore, the accuracy of the transfer function approximation will depend on the relative size of r and n.

The following two-step, closed-loop system identification procedure avoids the noise biasing problems of the process described above. We may express the system response as follows:

$$y = \frac{G}{1 + GC}r + \frac{1}{1 + GC}n.$$

Because the signals r and n are uncorrelated and the signal n has zero mean, averaging our system response yields

$$T_{yr} \triangleq \frac{G}{1 + GC} \cong \frac{y}{r}.$$

This approximation becomes an equality if the averaging is performed over an infinite time interval. Similarly,

$$u = \frac{1}{1 + GC}r - \frac{C}{1 + GC}n,$$

and averaging gives

$$T_{ur} \triangleq \frac{1}{1 + GC} \cong \frac{u}{r}.$$

Notice that both T_{yr} and T_{ur} can be estimated without a noise bias. Now, by simply dividing we may determine an unbiased estimate of the system response:

$$G = \frac{T_{yr}}{T_{ur}} \cong \frac{y/r}{u/r}.$$

In this lab, we will use the two-step, closed-loop system identification procedure described above to model a single input/single output path through the magnetic bearing. We will identify the path from OUT_{11} to OUT_{21} shown in Figure 4. Notice that the on-board controllers C_2, C_3 and C_4 are included in the system we are modeling.
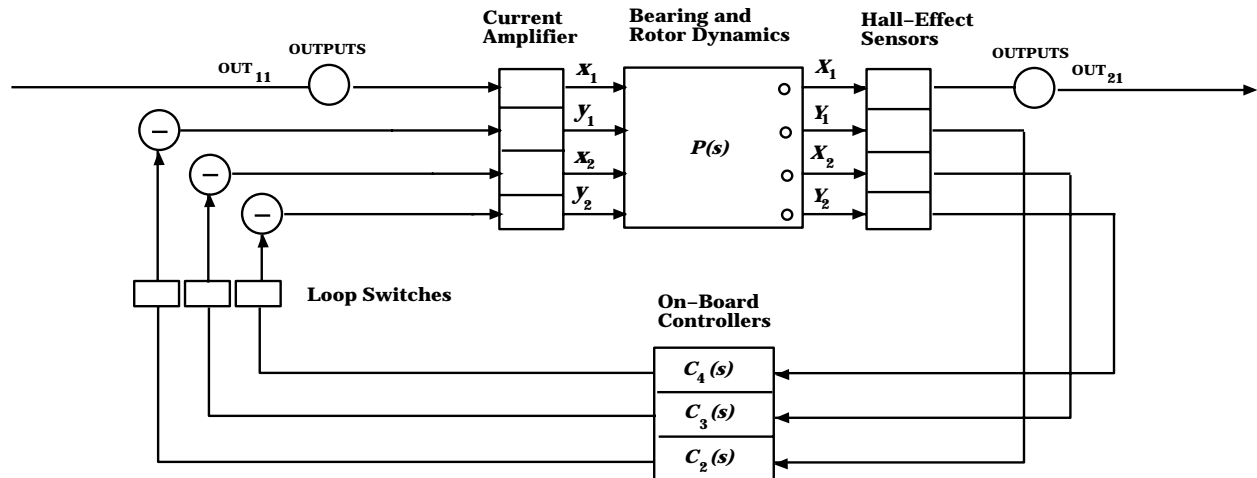

Figure 4: Magnetic Bearing System Path to Be Identified

In order to perform the first step of our identification process, we will do the following:

- Close all controller loops so that the rotor is levitated

- Drive the first system input IN_1 with the "Source" signal from the signal analyzer

- Connect the "Source" signal to "Channel 1" of the signal analyzer

- Connect the bearing output OUT_{21} to "Channel 2" of the signal analyzer.

In this configuration, the "Source" signal is treated as the system input and OUT_{21} is treated as the system output yielding the transfer function $T_{yr} = y/r$. This configuration is shown in Figure 5. Notice that this system is the same form as that of Figure 3 where G is the system of Figure 4 and $C = C_1(s)$.
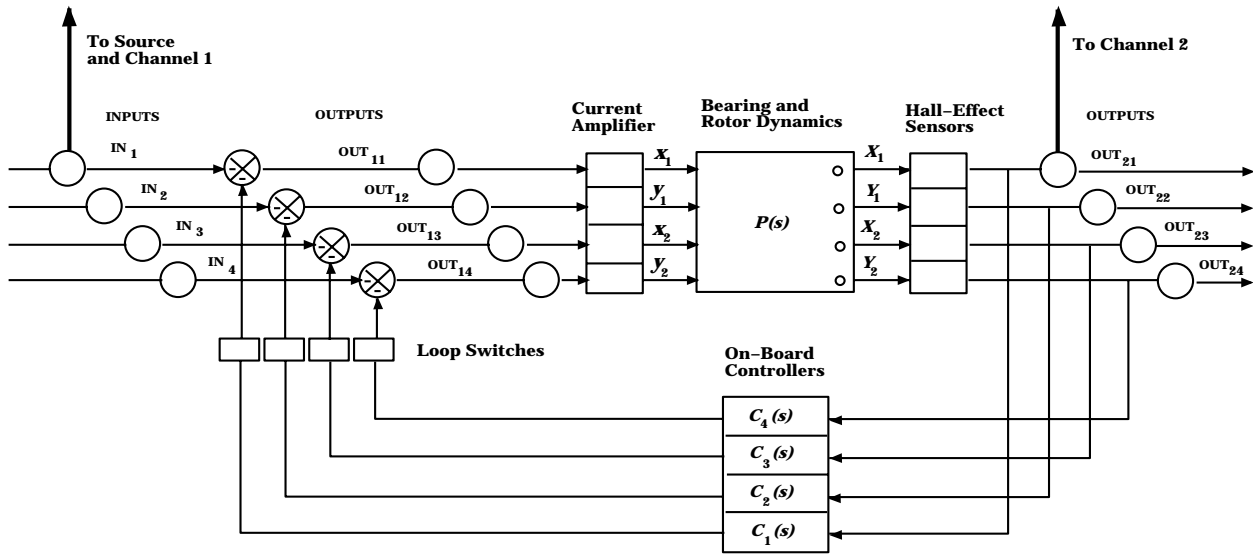

Figure 5: Bearing/Signal Analyzer Connections for Estimation of T_{yr}

Exercise 1: Connect the bearing system to the dynamic signal analyzer as shown in Figure 5 so that the transfer function T_{yr} can be estimated.

Next, we will set up the signal analyzer to perform a swept sine analysis of our system. The following exercises will describe this process step by step.

Exercise 2: For this exercise we will focus on the MEASUREMENT portion of the signal analyzer front panel. In order to select the swept sine function, first press the MEAS MODE key. Notice that the screen which appears is the default screen which also appears upon power up of the signal analyzer. Now using the soft keys (those keys to the immediate right of the screen), choose the SWEPT SINE function. Next, choose the SELECT MEAS key followed by the FREQ RESP soft key. We now choose a signal amplitude of 500 mV for the "Source" signal. Press the SOURCE key followed by the

SOURCE LEVEL soft key. Now type in 500 using the numerical keypad of the signal analyzer. After the number is entered, choose the soft key corresponding to millivolts (mV). Finally, we select the range of frequencies over which we will sweep. For this test, we will sweep from 10 Hz to 10 kHz. Press the FREQ key and then the START FREQ soft key. Now type in the number 10 followed by the soft key corresponding to Hz. Next, select the STOP FREQ soft key and choose 10 kHz. This sequence of steps can be represented concisely by the following table.

hard key	soft key	value	soft key
MEAS MODE	SWEPT SINE		
SELECT MEAS	FREQ RESP		
SOURCE	SOURCE LEVEL	500	mV
FREQ	START FREQ	10	Hz
	STOP FREQ	10	kHz

Table 1: Key Sequence for Initiating Swept Sine Mode

Exercise 3: We next configure our display so that a Bode plot representation of our system data will appear on the screen. We do this using the DISPLAY portion of the signal analyzer front panel. Choose the key labeled A. This makes the A portion of our display active. We now set up the A portion of our display to show the magnitude Bode plot. Choose the key MEAS DISP followed by the FREQ RESP soft key. Now choose the COORD key and select the MAG (dBm) soft key to choose a dB scale for the magnitude data. We now arrange to have the phase Bode plot appear on plot B. Press the key labeled B. Again, press the MEAS DISP key and then the FREQ RESP soft key. Press the COORD key again, but this time press the PHASE soft key to select the phase portion of the Bode plot. Finally, press the A&B key to display both the magnitude and phase portions of the Bode plot on the screen simultaneously. See Table 2 for a summary of the required key sequence.

Exercise 4: We are now ready to begin taking data. The measurement is begun by pressing the START key in the CONTROL portion of the signal analyzer front panel. If the data is taken at a constant amplitude over the entire frequency range, the system

hard key	soft key
A	
MEAS DISP	FREQ RESP
COORD	MAG (dBm)
B	
MEAS DISP	FREQ RESP
COORD	PHASE
A&B	

Table 2: Key Sequence for Bode Plot Display

may become unstable at the frequency ranges corresponding to the rotor resonances (around 800 Hz and 2000 Hz). Thus, when the sweep nears these frequencies, the amplitude may have to be reduced. Typically, 200 mV or 100 mV will yield good results. To reduce the amplitude, choose the PAUSE/CONT key. This will pause the sweep. Then select the SOURCE key followed by the SOURCE LEVEL soft key. A new signal amplitude may now be entered. The frequency sweep may be continued by pressing the PAUSE/CONT key once again. If the rotor hits one of its physical limits at any time while data is being collected, the data will be corrupted and the test must be run again. If the bearing becomes unstable, it may make a loud noise as it hits its physical stops, but this is not dangerous. Just turn off the bearing system and pause the signal analyzer sine sweep. Then the bearing system can be turned on again and the test run again by simply pressing the START key.

In the next few exercises, we will save the data collected in the previous exercises onto the signal analyzer hard disc, transfer the data to a PC, and then change its format to the type used by MATLAB. It is assumed that the user has access to the Standard Data Format Utilities software available from Hewlett Packard and that the PC to be used is installed with an IEEE 488 port card and driver for making the parallel port connection with the signal analyzer.

Exercise 5: To save data to the signal analyzer hard disc, the parallel port connection between the signal analyzer and the hard drive must be in place and the hard drive must be turned on. Select the DISC key from the HP-IB portion of the signal analyzer

front panel. Choose the SAVE FILE soft key and type in a file name using the blue letters located to the bottom right of each front panel key. Finally, press the soft key labled ENTER.

hard key	soft key	text	soft key
DISC	SAVE FILE	FILENAME	ENTER

Table 3: Key Sequence for Saving Data to Hard Disc

In the second step of the identification process, we will estimate the transfer function $T_{ur} \cong u/r$. To properly configure the bearing system for this part of the experiment, we must:

- Close all controller loops so that the rotor is levitated

- Drive the first system input IN_1 with the "Source" signal from the signal analyzer

- Connect the "Source" signal to "Channel 1" of the signal analyzer

- Connect the bearing output OUT_{11} to "Channel 2" of the signal analyzer.

Figure 6 shows the signal analyzer connections described above. Again, note the similarity with Figure 3.

Exercise 6: Repeat Exercises 2-5 to perform a swept sine analysis of the tranfer function T_{ur} and save the response data on the signal analyzer hard disc.

For the following discussion, it is assumed that all of the Standard Data Format Utilities software has been loaded into the root directory of the C drive of the PC in use. It is also assumed that the user saved the data into a file entitled `filename` on the signal analyzer hard disc and wishes to transfer it to the PC subdirectory `C:\data`.

Exercise 7: Make sure that the signal analyzer hard disc is connected to the PC via a parallel port connection. Type the command `lif scan` to list the name of the HP hard drive. (For the purpose of illustration in this exercise, we assume the hard drive to be
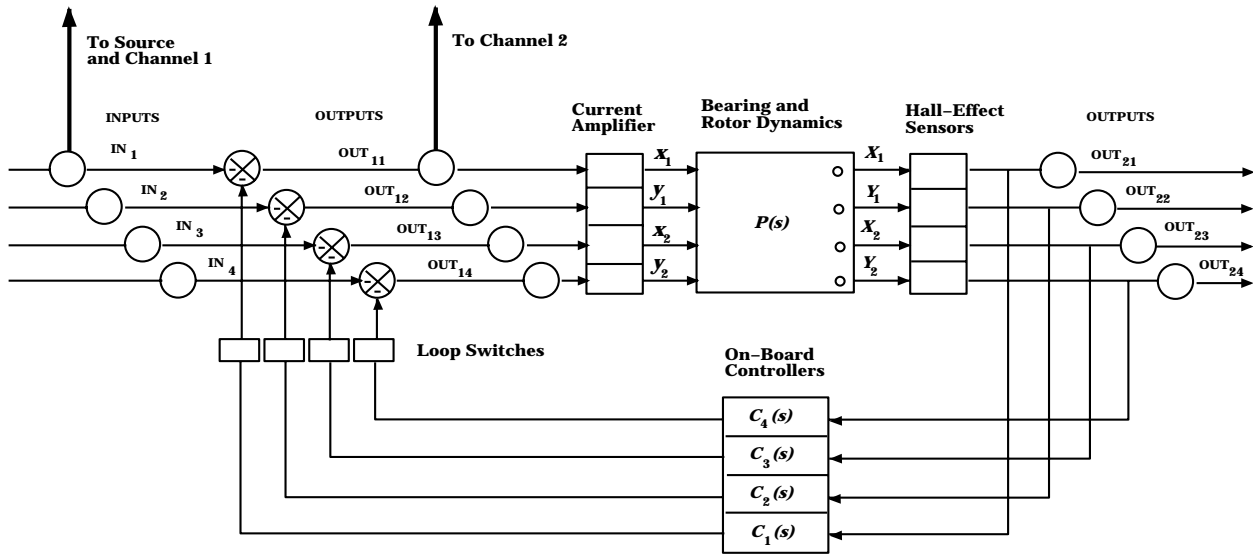
10

Figure 6: Bearing/Signal Analyzer Connections for Estimation of T_{ur}

named `101:`) To list the files currently saved on the signal analyzer hard drive, type the command

```
lif dir 101:
```

from the C drive of the PC. Notice that a file saved as `filename` will be listed as `dafilename` when the `lif dir` command is used. Now copy the two files containing the frequency responses T_{yr} and T_{ur} to the PC by typing the command

```
lif cp 101:  dafilename C:\data\ filename.63
```

for each file.

Exercise 8: We now use the Standard Data Format Utilities to convert the file from the signal analyzer format to the MATLAB workspace format. For each file, type the command

```
63tosdf C:\data\filename.63 C:\data\filename.sdf
```

Then for each file, type the command

```
sdftoml C:\data\filename.sdf C:\data\filename.mat /x
```

The extension `/x` causes the conversion to include a vector of the frequencies corresponding to each data point. Each resulting MATLAB file will have two vectors. The first vector, `o2i1`, is the vector containing the complex numbers which are the corresponding transfer function evaluation at each frequency point. The second vector, `o2i1x`, is the vector of frequency points in Hz.

2.3 Finding a Model to Fit the Data

In order to manipulate the data we have collected, we must first load it into a MATLAB workspace. Assume for the purpose of illustration that the transfer function data corresponding to T_{yr} is contained in a vector labled `tyr` and similarly, the data corresponding to T_{ur} is contained in the vector `tur`. We may obtain our transfer function estimation $G \cong T_{yr}/T_{ur}$ by an element-wise divide of the two transfer function vectors as follows:

$$g\texttt{=tyr ./ tur}$$

In order to determine a transfer function fit to the system data `g`, we will be using the MATLAB function `fitsys` contained in the μ-TOOLS toolbox. For this function, we need to define a matrix `resp` which is a combination of the frequency response data for our system and the frequency points in radians per second at which the data was collected. The vector of frequency points in rad/sec `rps` and the matrix `resp` are created as follows:

$$\texttt{rps=2*pi*o2i1x}$$

$$\texttt{resp=vpck(g,rps)}$$

We will also define a variable `ord` equal to the order of the transfer function fit we desire. Finally, we define a vector `wt` which is a frequency weight allowing the user to emphasize certain frequencies over others in the fit. The vector `wt` is equal in size to the frequency response data vector `g` and each element in `wt` determines the relative importance of the corresponding data point in determining the transfer function fit. Now we are ready to use the `fitsys` function to find a model fit to the data. `fitsys` generates a system matrix `sys` which can be decomposed or unpacked into a state space model for the system. The syntax of the `fitsys` function call is as follows:

$$\texttt{sys=fitsys(resp,ord,wt)}$$

A state space representation of our data fit model can be written:

$$\dot{x} \;=\; asys\,x + bsys\,\mathrm{OUT}_{11}$$

$$\mathrm{OUT}_{21} \;=\; csys\,x + dsys\,\mathrm{OUT}_{11}$$

where the system matrices $asys, bsys, csys$, and $dsys$ are determined from the system matrix `sys` as follows:

$$\texttt{[asys,bsys,csys,dsys]=unpck(sys)}$$

This state space model can then be converted into a transfer function model with the following command:

$$[\texttt{num,den}]\texttt{=ss2tf(asys,bsys,csys,dsys)}$$

where `num` corresponds to the numerator of the transfer function and `den` corresponds to the denominator of the transfer function. See the MATLAB manual or help menus for detailed descriptions of these functions. In our case, we will desire a transfer function fit of order 9 and we will want to emphasize the frequencies corresponding to the system resonances located near 800 Hz and 2000 Hz because these are so critical to the design. Therefore, we choose `ord=9` and `wt` as a vector of size equal to `g` with a 1 at every position corresponding to frequencies not involving the resonances and some bigger number M at the resonance frequency points. The size of M can be determined by the user in order to accurately model the total frequency response characteristic.

In order to determine the best transfer function fit to the data, we will need to plot the Bode plot of the data vs. that of the transfer function fit. The following sequence of MATLAB commands will plot the Bode plots together for comparison.

$$\texttt{resp2=frsp(sys,rps)}$$

$$\texttt{vplot('bode\_g',resp,resp2)}$$

Exercise 9: Load the two data files corresponding to T_{yr} and T_{ur} into MATLAB and obtain the system response vector `g`. Using the `fitsys` function in μ-TOOLS, find a 9th order transfer function model of the frequency response data contained in `g`. Choose your weight to achieve the "best" trade-off between fitting the overall frequency response and accurately modeling the resonances.

Save the data vector `g` and frequency vector `rps` for use later when designing the controller for this system. You can save them in a MATLAB data file by typing the following:

$$\texttt{save system g rps}$$

This command will create the file 'system.mat' containing the vectors `g` and `rps`. They can be loaded into another MATLAB workspace by simply typing

$$\texttt{load system}$$

Version	Date	Modifications
3	4/24/96	MATLAB code for creating vector of frequency points **rps** added on p. 12
		MATLAB code for creating frequency response vector **resp** corrected on p. 12
		Instructions and MATLAB code for plotting the Bode plots of data vs. transfer function fit added
		Instructions and MATLAB code for saving frequency response data added on p. 13
		Minor notational and formatting modifications made